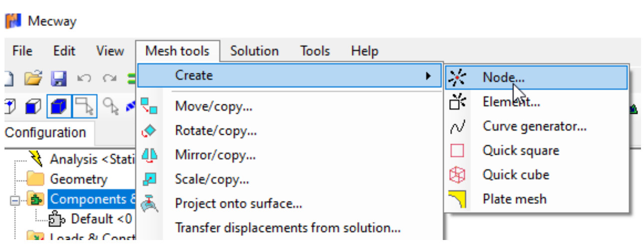
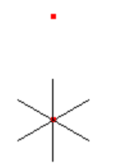
Mecway - Model Creation

For this case, we want to start from scratch and will not import a CAD model. This is just one approach to make a beam that is made of solid elements.

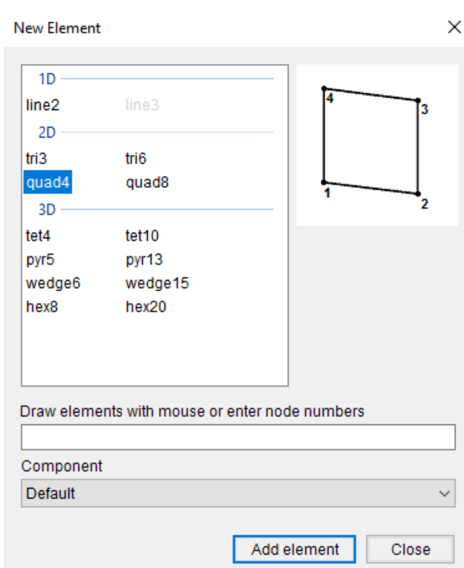
If you select the "Components and Materials" section, "Mesh Tools" will be available. You can then create 4 nodes that define the square cross section…

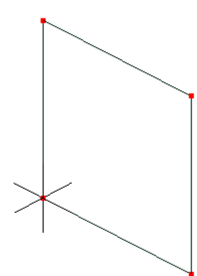


Nodes in red



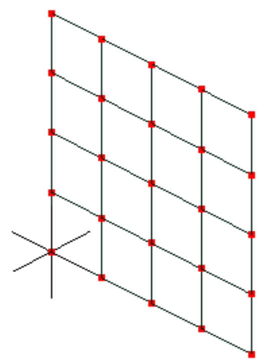
Next, select "Mesh Tools/Create Element". Then click the 4 nodes you created in the prior step.





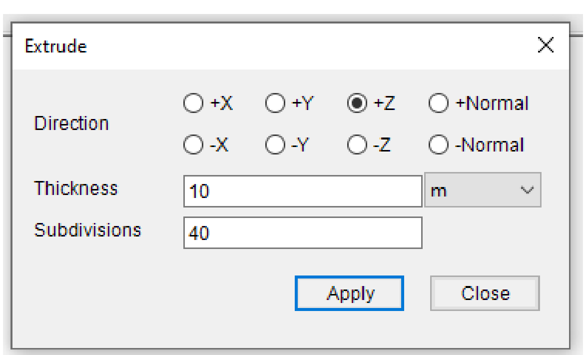
Mecway - Model Creation

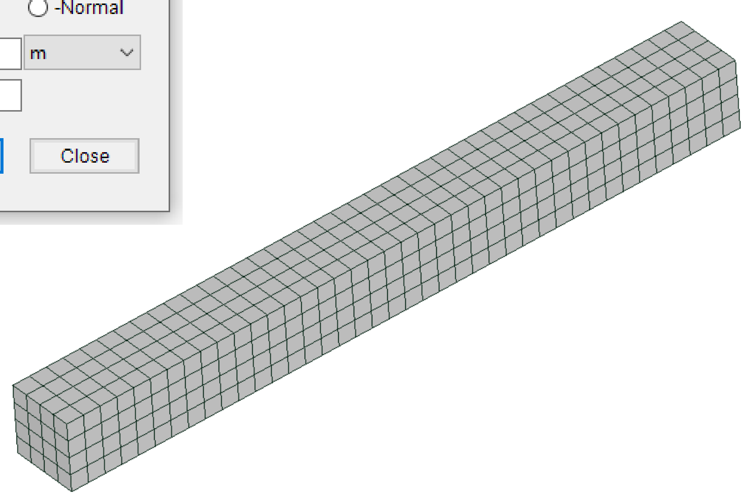
Use “Mesh Tools/Refine/x2” and repeat this…



Ensure the “filter” is set for faces or elements and select all the elements. Then use “Mesh Tools/Extrude…” as follows. The solid elements (brick elements) will be created.

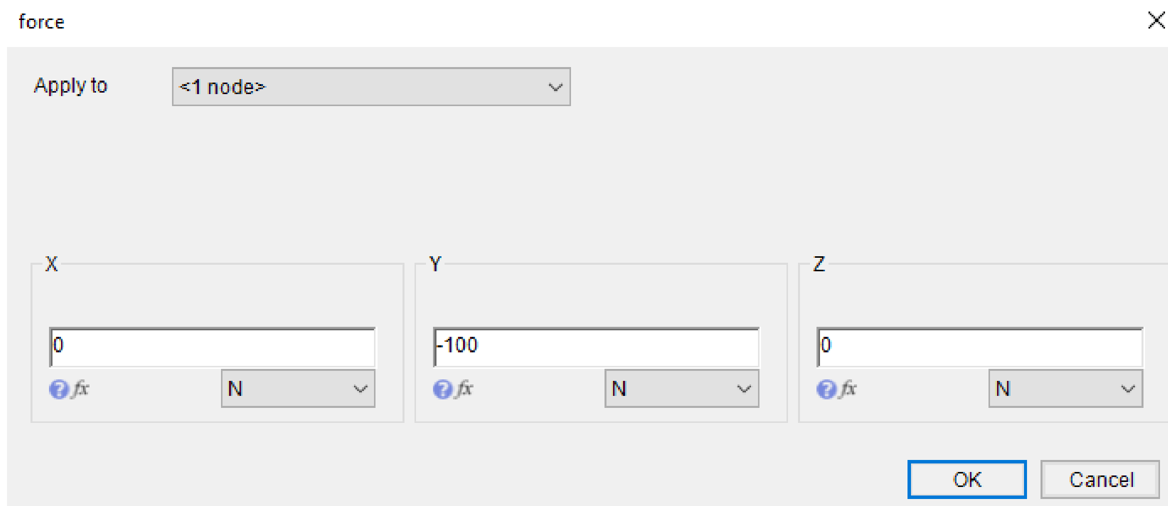
Picking “filter” menu



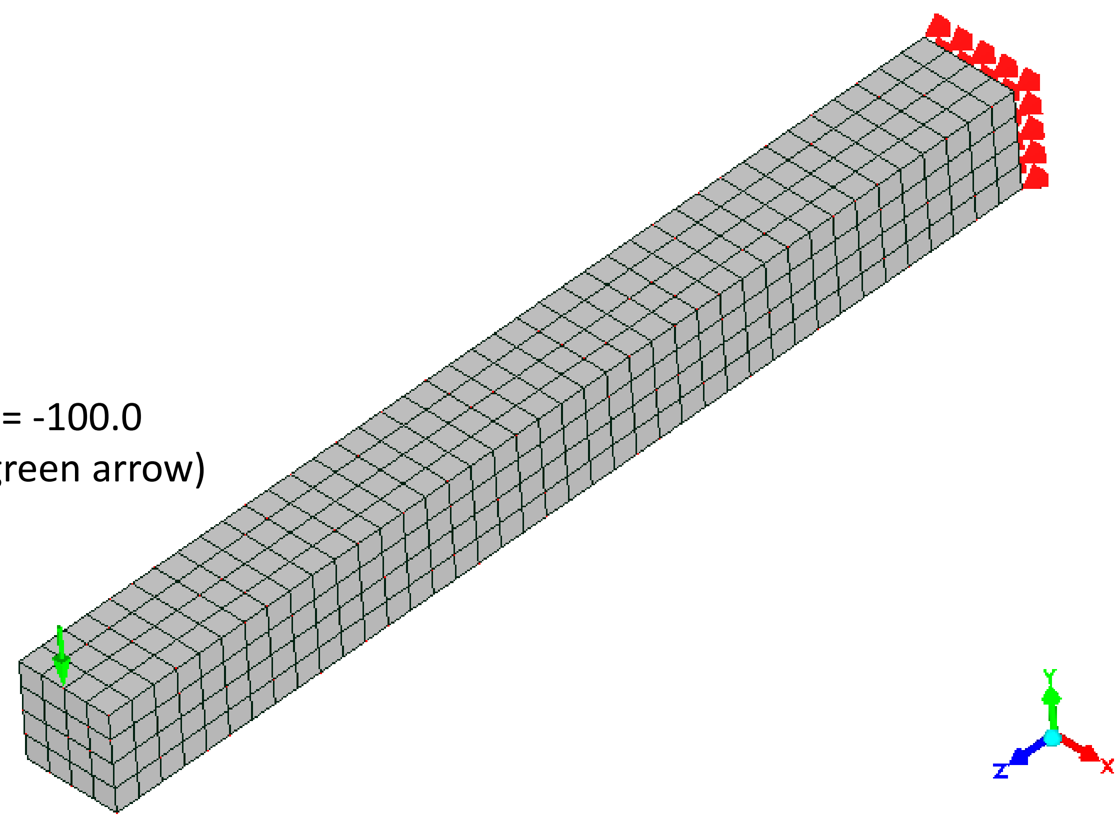


Boundary Conditions

A cantilever load is shown below. In Mecway, right click on the "Loads & Constraints" section to add the force.



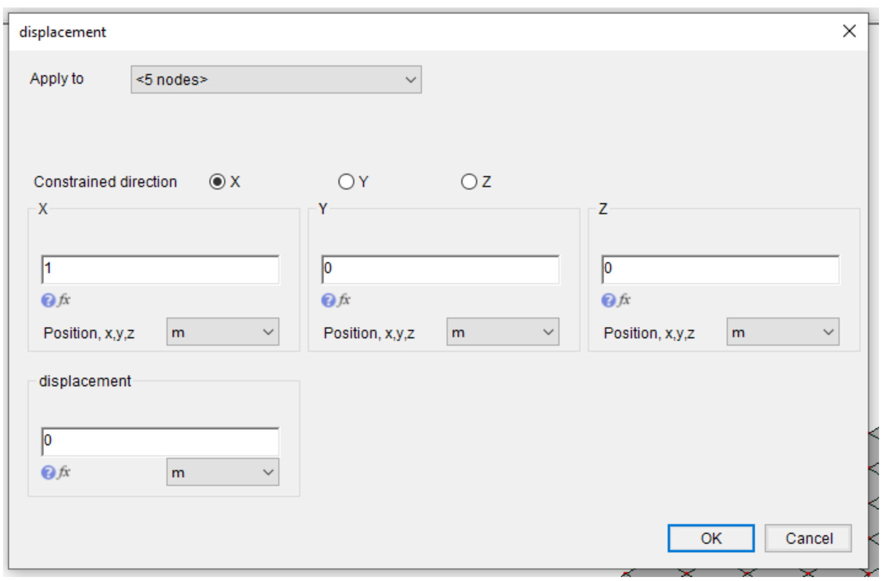
P = -100.0
(green arrow)



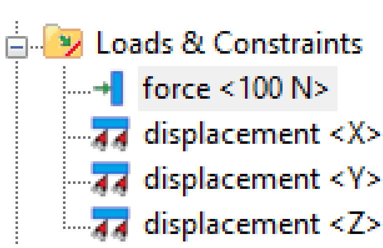
Boundary Conditions

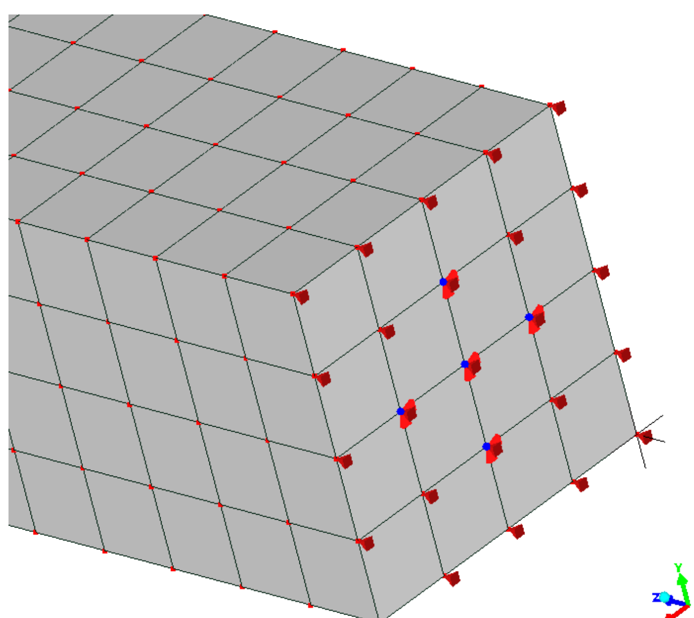
At the wall, all Z-displacements are constrained (25 nodes).

However, only the nodes in blue (5 nodes) are constrained in the X- and Y-directions. These prevent rigid body motion, but avoid Poisson effects so the results can be compared to the classical solution. These are applied as 3 separate items (shown below).

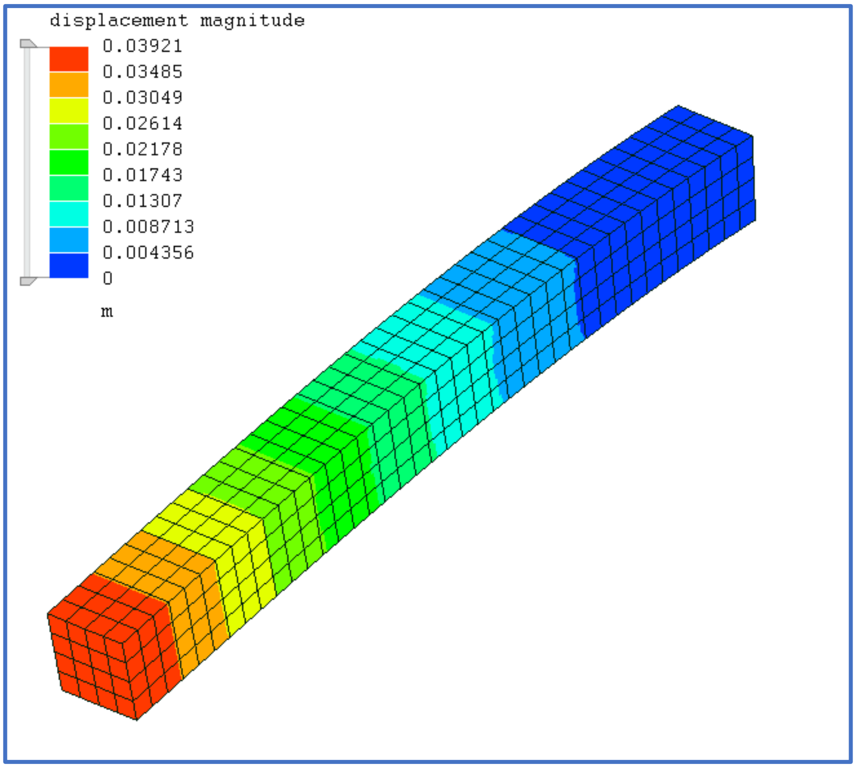


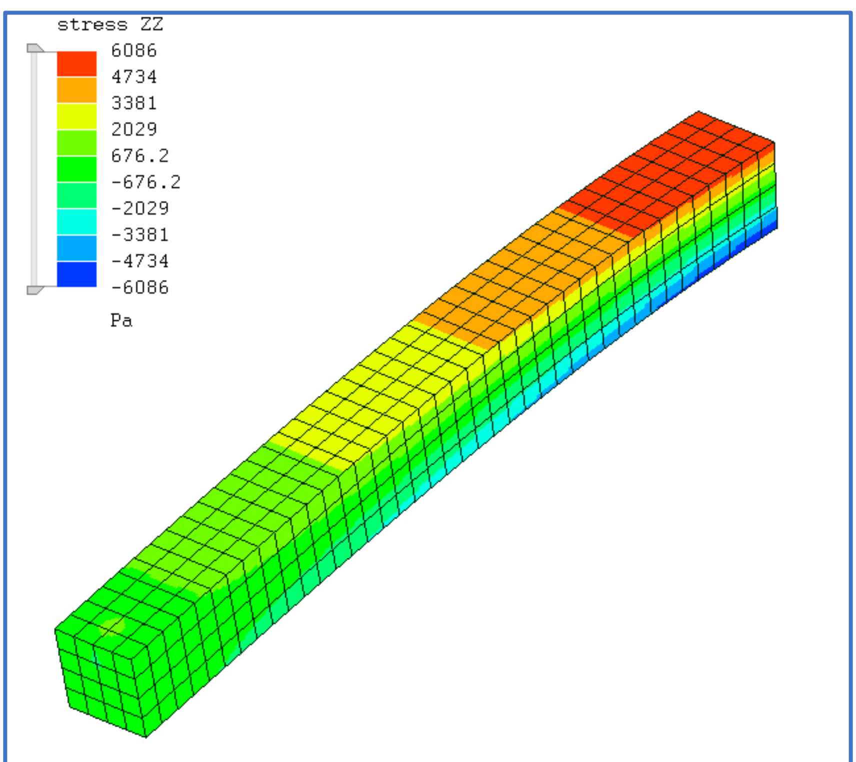
Entry for the X-displacement





Results – Mecway Internal Solver





Calculations

Classical Solution

$P := 100$ $\quad$ $E := 10 \cdot 10^{6}$ $\quad$ $b := 1$

$L := 10$ $\quad$ $t := 1$

$M := P \cdot L$ $\quad$ $M = 1000.0000$ $\quad$ $I := \frac{1}{12} \cdot b \cdot t^{3}$ $\quad$ $I = 0.0833$

$\delta := \frac{P \cdot L^{3}}{3 \cdot E \cdot I}$ $\quad$ $\delta = 0.0400$ $\quad$ Deflection at the end

$\sigma := \frac{M \cdot \left(\frac{b}{2}\right)}{I}$ $\quad$ $\sigma = 6000.0000$ $\quad$ Stress at the wall (upper surface)

Mecway Internal Solver

Displacement - less than 2% error

Stress - less than 2% error

MYSTRAN Solver (Images Shown Later)

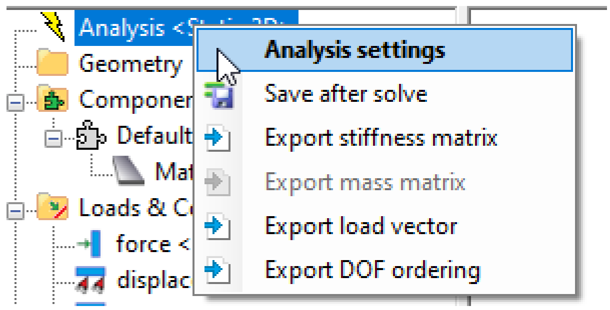
Displacement - less than 1% error

Stress – 25% Error (see later slides). This is because MYSTRAN/pyNastran report the centroid result of the element (not the maximum stress at the corner of the element). Mecway appears to report the maximum corner stress.

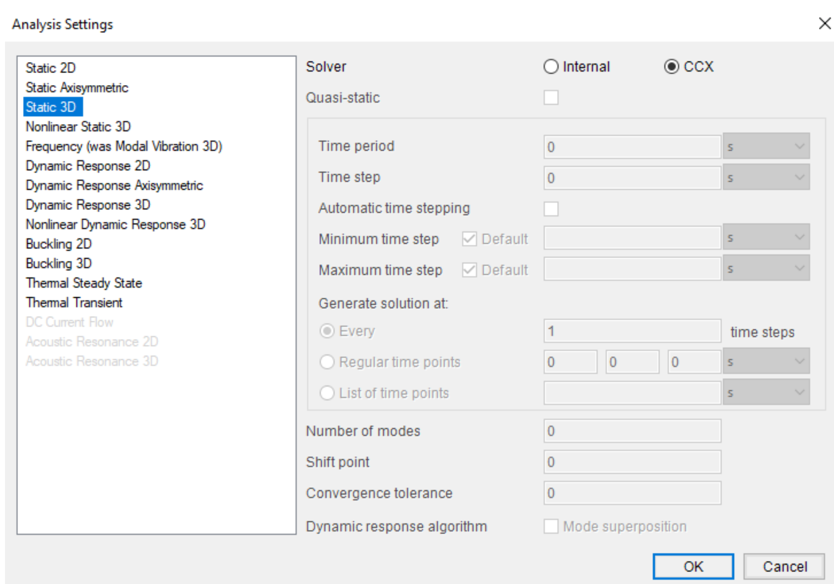
NOTE: There results are not necessarily useful to determine if one program is more “erroneous” than another (the mesh is not necessarily converged). This is mainly a “sanity check” to ensure the entire process is correct and the model is acceptable for study. A result within about 5-10% would have been considered acceptable since this not a “fine” mesh.

Mecway to CalculiX INP

Rather than use the Mecway internal solver, right click Analysis and select “Analysis Settings”



Then select CCX (CalculiX CrunchiX). This will allow you to create the CCX input deck (INP file).



Then click “Save As” and select “CalculiX input deck (.inp)”

NOTE: All units will be in MKS and Mecway is not the typical “unitless” pre/post processor. If you want want to work in IPS, just use the MKS system and disregard what Mecway says the units are.

CalculiX INP to MYSTRAN BDF

The pyNastran program is a post processor for NASTRAN (and by extension MYSTRAN). While it has some preprocessing capabilities, it is primary used for postprocessing.

Because there are limited options for native NASTRAN preprocessor support, the pyNastran author created an ABAQUS to NASTRAN converter. By extension, CalculiX INP decks can be used for this. This is an advanced converter that can link the properties, boundary conditions, etc to the elements. It is not a simple “line for line” converter. However, there will be limitations because ABAQUS/CalculiX does not support all the elements than NASTRAN does (for example the BUSH element). However, the basic elements (solids, shells, beams, rods, springs - translational) are supported.

The converter is part of pyNastran and the install instructions are here:

https://github.com/MYSTRANsolver/MYSTRAN_Resources/blob/main/Postprocessor/pyNastran/Windows_Install.txt

NOTE: The converter is still in a state of development. Contact the author if problems arise.

NOTE: As of 1/4/2024, you must go through pyNastran to use the converter. At a later date, the converter will have a stand-alone option.

NOTE: This entry needs to be resolved.

```
*EL FILE
S,NOE
```

Why MYSTRAN?

With the existence of the Mecway internal solver and the CalculiX solver supported in Mecway, why would one want to use MYSTAN? After all, its significantly more effort to use MYSTRAN in this manner. For some (and perhaps many) cases, MYSTRAN is not the best choice. However, for some cases, you may find MYSTRAN to be preferable.

CalculiX vs MYSTRAN

Unlike CalculiX, all of the MYSTRAN elements are “classical”. CalculiX uses classical elements for solid elements, but the shells, beams, springs, etc. are expanded to solid elements internally. This can create various postprocessing challenges and also challenges with aspect ratio, MPC’s etc.

Mecway Inteneral Solver vs MYSTRAN

MYSTRAN is very extensive FEA solver, that has a nearly 300 page user manual. You have full control over the solver inputs and all of “tweaks” and adjustments to the deck are possible.

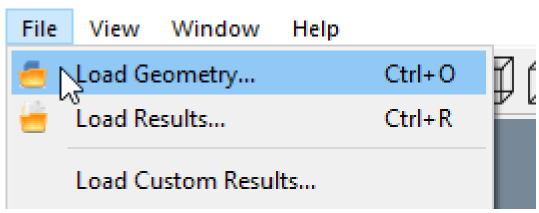
Some considerations are:

- MYSTRAN I/O is highly compatible with NASTRAN. So if you are already familiar with NASTRAN, the learning curve will be minimal.
- You can manually add “special” elements such as BUSH and CSHEAR
- MYSTRAN is in the process of validating models compared to other NASTRAN version (hundreds and eventually thousands of decks will be verified for each build)
- MYSTRAN is open source
- MYSTRAN will soon be able to create advanced CSV files for post processing.
- MYSTRAN can handle multiple load cases
- You may find the pyNastran postprocessor to be advantageous (see the following slides)

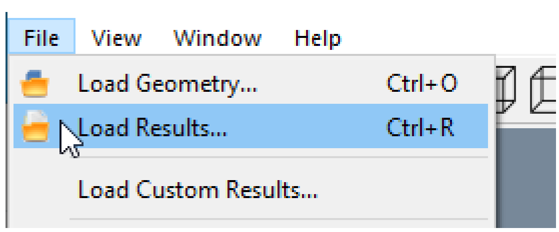
PyNastran - Postprocessing

After you run the model in MYSTRAN, you need to post process the results. Unfortunately, Mecway can not post process the results, but pyNastran can.

You first need to load the "geometry" (DAT or BDF). Note, although this is called "geometry", its may also load in material properties, element properties, boundary conditions, etc. into pyNastran (depends on preferences).

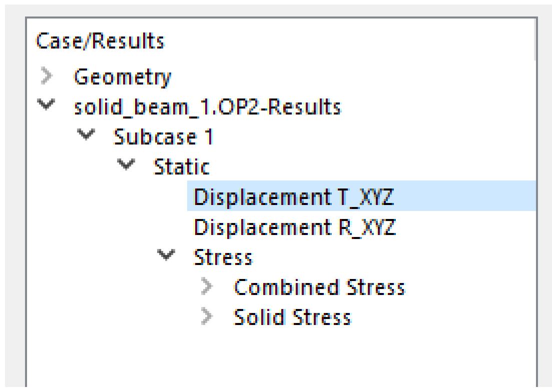


Then, load the results. This is the OP2 file that was generated by MYSTRAN.

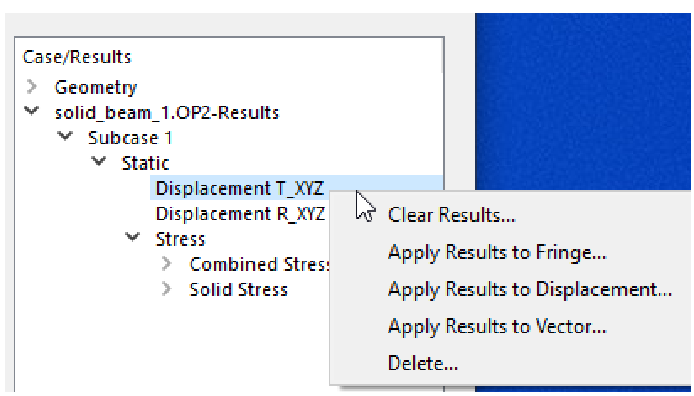


PyNastran - Postprocessing

You can now access the results. You can drill down like such to see the displacement.



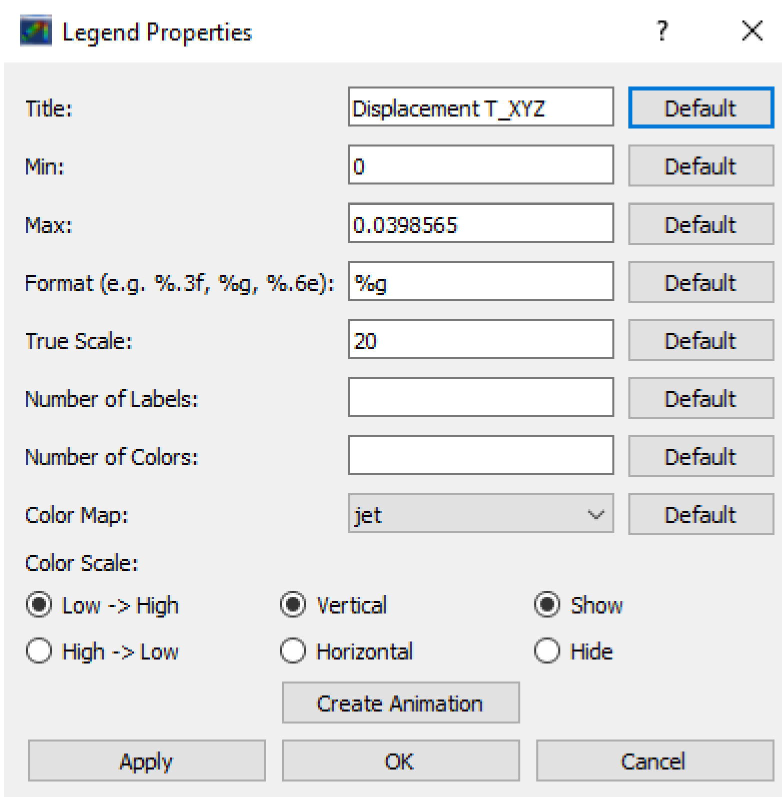
You can toggle between a fringe only result and a fringe + displacement result (called "Apply Results to Displacement…")



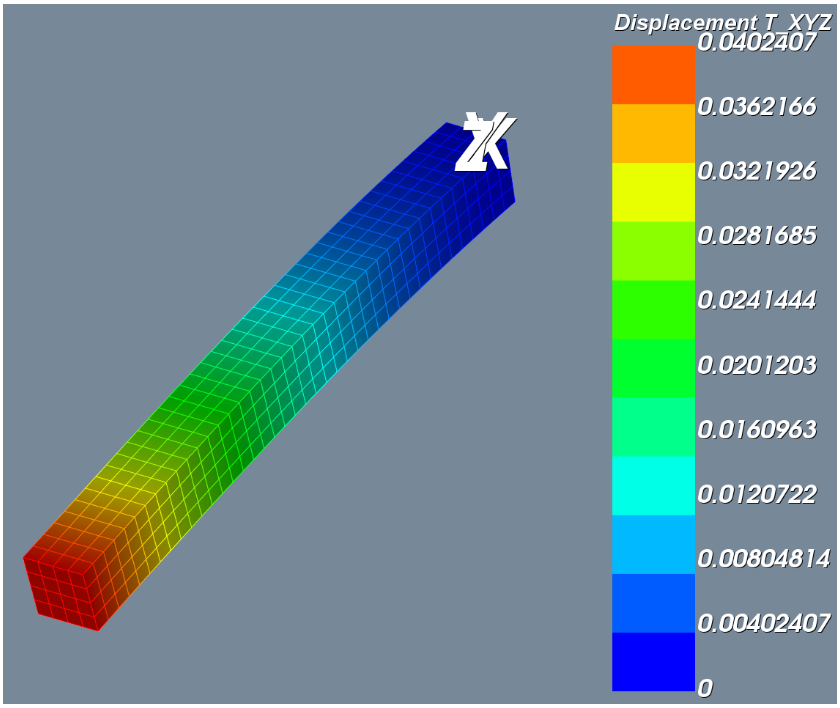
PyNastran - Postprocessing

You can adjust the scale by selecting “View/Modify Legend”. The following pop up box will show.

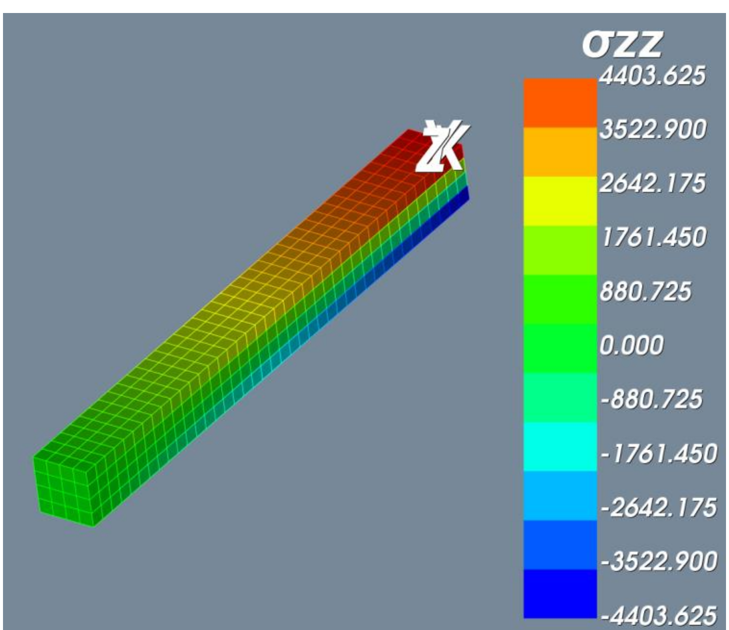
You can also animate the displacement from this window.



PyNastran – MYSTRAN Results



MYSTRAN = 0.0402
NX = 0.0405 (ref)
1% different



MYSTRAN (solid centroid) = 4403
NX (solid centroid) = 4454 (ref)
1% different

These results are for the centroid of the solid element (not the maximum corner stress).

Note that from the NX F06 file, the maximum corner stress is 6023, which is accurate to within 1%.